Renewable and Sustainable Energy Reviews 13 (2009) 1620–1627

Contents lists available at ScienceDirect

Renewable and Sustainable Energy Reviews

journal homepage: www.elsevier.com/locate/rser

In-cylinder fluid flow, turbulence and spray models—A review

Syed Ameer Basha *, K. Raja Gopal

Jawaharlal Nehru Technological University, India

ARTICLE INFO

Article history:
Received 22 August 2008
Accepted 30 September 2008

Keywords:
In-cylinder fluid flow
Modeling
Spray
Turbulence

ABSTRACT

This article is a literature review of use of computational fluid dynamics (CFD) codes to model the in-cylinder fluid flow, turbulence and spray characteristics. This study is based on the reports of about 60 scientists, who published their results between 1978 and 2008. Most of the scientists and researchers used CFD codes to analyze the models under simulation conditions and compared these simulated results with experimental results. Some scientists reported that different engines exhibit different behaviors with similar fuel sprays and Re-Normalized Group (RNG) k-ε model is the best applicable turbulence model for engine simulation. The KIVA code is widely used for model development in academia due to the availability of the source. However, its capability for resolving complex geometries is limited. On the other hand, other commercial CFD codes such as STAR-CD, FIRE, VECTIS and FLUENT are frequently used by the industry due to their superior mesh generation interfaces and because of their available user support. Some scientists combined STAR-CD and KIVA code for the engine simulations but they concluded that, it would be preferable to implement the advanced submodels directly into one commercial code for engine simulations.

Contents

1. Introduction

Fluid flow in an internal combustion engine presents one of the most challenging fluid dynamics problems to model. It is because the flow is associated with large density variations. Further, the fluid motion inside a cylinder is turbulent, unsteady, cyclic and non-stationary both spatially and temporally. The combustion characteristics were greatly influenced by the details of the fuel preparation and the distribution of fuel in the engine cylinder, which was mainly controlled by the in-cylinder fluid dynamics. Further, fuel injection introduces additional complexities. It was mainly due to the difficulties in understanding the physics of two-phase flows that vary spatially and temporally from very dense to fairly dilute. Pollutant emissions were controlled by the details of the turbulent fuel–air mixing and combustion processes. So, a detailed understanding of these processes is required to improve performance and reduce emissions without compromising fuel economy.

2. In-cylinder fluid flow models

The role of in-cylinder air motion commences from the very start of the engine cycle. During the intake stroke, the incoming air generates flow structures with large-scale turbulent motions within the cylinder, which in turn determines the extent of mixing

* Corresponding author.
E-mail addresses: sdameer2001@yahoo.co.in (S.A. Basha), rajagopalmail@rediffmail.com (K. Raja Gopal).

1364-0321/$ – see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.rser.2008.09.023

between the fresh charge and the residuals, as well as internal and external heat transfer rates. The flow field plays a vital role in charge preparation and conditioning for ignition. During the combustion processes, the flow field continues to exert its effect on heat transfer, peak flame temperatures and thermal stress levels. In the post combustion period, the nature of fluid motion determines the level of pollutant emissions. The flow field also determines the cyclic variation exhibited by it and all other processes, which it influences.

Margery investigated the affect of intake duct length on volumetric efficiency and the in-cylinder flow field in a four-stroke single cylinder direct injection (DI) diesel engine motored at 1000–3000 rpm. Three lengths of straight duct upstream of the helical inlet port of the engine are considered and measurements of instantaneous mass flow rate and pressure drop across the port are reported as a function of duct length and engine speed [1]. Zhang et al. analyzed the combustion phenomenon of a direct injection diesel engine by using the cross-correlation method and two-color method to measure the combustion flame motion and the flame temperature, respectively by processing the high-speed in-cylinder photographs. The effects of engine parameters such as pumping rate, injector nozzle hole size and injection timing on combustion processes, particularly on flame motion and flame temperature were studied [2].

Chen and Dent presented a combined experimental and computational study of steady flow through a curved inlet port. Flow structures affected by valve lift and port shape are predicted by the computational fluid dynamics (CFD) code STAR-CD. Local static surface pressure maps are obtained experimentally [3]. Kong et al. developed an image acquisition and processing camera system to acquire luminous flame images from the combustion chamber of a single cylinder heavy-duty diesel engine. Experiments were conducted both single and split injection cases at 90 and 45 MPa injection pressures at 3/4 load and 1600 rpm with simulated turbo charging. The experimental results are compared to numerical computations using a version of KIVA-II code with improved spray, ignition, combustion and emission models. Good levels of agreement are obtained with measured cylinder pressures, heat release rate, ignition timings and locations, and flame shapes [4].

Zhang et al. carried out an experimental study aiming to investigate the effects of combustion chamber geometry on combustion process in an optically accessible direct injection diesel engine. Flame movement behaviors such as the distribution of flame velocity vectors and the averaged flame velocity inside and outside the combustion chamber are measured by means of cross-correlation method. The combustion geometries considered were dish and re-entrant chambers [5]. Stephenson and Rutland assessed the differences in the in-cylinder flow field present during combustion resulting from different intake flow configurations and compared with the significance of spray–wall interaction effects on combustion and emission performance. The key differences observed in the flow field at the onset of fuel injection were turbulent kinetic energy and the magnitude of the swirl flow. Quantities such as the tumble and swirl flows and crossing frequency length scales proved to be helpful in understanding the development of the flow field during intake and compression. By the time, the fuel injection began, most of the tumble flow angular momentum had been dissipated or converted into swirl flow. The most significant differences in the injection flow field were in the turbulence intensity and swirling flow, both of which maintained differences in intensity developed early in the intake flow throughout the intake and compression strokes. At idling operation, the differences generated by intake flow were found to be considerably less important than the effect of fuel spray–wall impingement in respect of pollutant formation. At 75% load, the effects of intake flow were comparable to those of wall interaction [6].

Stephenson et al. conducted a parametric study on the effects of swirl, initial turbulence, oxygen concentration and ignition delay on fuel vaporization, mixing and combustion process. The effects of intake flow field was further investigated by varying the geometry of the intake ports and intake runners in a model dual port direct injection diesel engine in KIVA simulations. The code solves the three dimension conservation equations and uses an improved RNG k-ε turbulence model [7]. Catania and Spessa used an advanced hot wire anemometer (HWA) technique for air velocity measurements in the combustion chamber of an automotive diesel engine with a slightly re-entrant cone shaped in-piston bowl and one helicoidal intake duct under motored conditions. The in-cylinder flow data are acquired in a crank angle interval ranging from 30 degrees before intake top dead centre (TDC) to 60 degrees after compression TDC, so as to investigate the direct and reverse squish flows produced by the bowl, an intense and high frequency turbulence production occurred during the induction stroke which was ascribed to the secondary and main intake jet flows. Over the end of this stroke and during the start of compression, the turbulence intensity and frequency, the mean flow and its cyclic fluctuation showed to decay at crank angles and rates that increased with the engine speed. During the main part of compression, these quantities tended towards almost stationery distributions with a fluctuating pattern at the highest speeds and with a maximum in the mean flow at the lowest speeds over mid compression. These trends were most likely due to flow unsteadiness, compression and stratification effects. At the end of compression, a further decay occurred in the mean flow and turbulence intensities up to the onset of the direct squish flow where an increase of intensity and frequency are generally observed. During the investigated part of the expansion stroke, a strong growth in the mean flow and in both turbulence intensity and frequency are detected at all speeds. This is ascribed to the conical in-piston bowl nozzle effect of intensifying the reverse squish jet flow and its interaction with swirl [8].

Leylek et al. examined steady-stake intake region flow for low, medium and high valve lift cases in a caterpillar diesel engine, with the intent of investigating total pressure losses. Computationally predicted loss pockets are identified and presented thoroughly for high valve lift case. Result shows upto 30% of total pressure loss occurring in the upstream of valve clearance region and highest losses occurring in the valve clearance region [9].

Bo et al. assessed the accuracy of prediction of the air motion and spray characteristics in a direct injection diesel engine with the CFD code. The Laser Doppler Anemometry (LDA) measurements of velocity field are obtained for the induction and compression strokes upto the time of ignition. The predictions of gas velocities spray penetration and droplet velocities for the most part reproduce the experimental data. However, further evaluation of the spray modeling particularly in respect of the effects on fuel vapor and gas temperature needed [10]. Chen et al. simulated the transient flow for the induction stroke using STAR-CD version 2.21 software. The standard k-ε model was employed to represent the fully turbulent flow with the 'wall function' for description of boundary layer behavior [11]. Auriemma et al. made in-cylinder measurements under motoring conditions in a light duty diesel engine equipped with a re-entrant bowl in piston combustion chamber. Tangential and radial components of the air velocity are acquired over a crank angle range of 90° above TDC. The mean motion, integral time scale and Reynolds shear stress are estimated by applying an ensemble averaging technique [12]. Beard et al. carried out a combined experimental and computational investigation of the flow

fields in a direct injection diesel engine. Simulations were performed with a modified version of KIVA II code taking into account, the optical piston elasticity due to the high pressure [13]. Brunt and Platts examined and quantified the main sources of error in gross heat release calculations using the traditional first law heat release model. The most significant errors were due to incorrectly assigned ratio of specific heats and charge to wall heat transfer rates. An alternate heat release model, the polytropic index first law model was developed and compared with experimental results obtained from a direct injection diesel engine [14]. Okazaki et al. studied the improvement of CFD approach for practical designing of the direct injection diesel intake and combustion system. A tetrahedron cells model was effective to get a relative evaluation of steady-state flow. The flow behavior of the actual engine operating condition could be predicted by the transient analysis that could be used for quantitative comparison of intake port performance [15].

Bianchi et al. investigated the influence of different initial condition procedures on combustion and emissions predictions in small-bore high-speed direct injection diesel engine. The analysis was performed by using the STAR-CD code for the intake stroke calculation and KIVA-II for the compression stroke and combustion simulation. Analysis of the evolution of fluid dynamic parameters has revealed that a detailed definition of the initial conditions is required to properly predict the mean and turbulent flow field at the time of injection near TDC [16]. Lin carried out numerical analysis of air motion in three different shapes of combustion chambers (central projection type, shallow W type and pataloid type) in order to reduce the NO_x and smoke emissions. The experimental study on a single cylinder diesel engine with central projection chamber showed that this system exhibited the rapid combustion process and short main combustion period. Retardation of injection could realize a large reduction in NO_x and slight rise in fuel consumption and smoke emission [17].

Reitz et al. examined in detail the velocity field in a direct injection diesel engine employing both in-cylinder velocity measurements and results of multi-dimensional calculations using KIVA 3V code [18]. In-cylinder fluid velocity was measured by Miles et al. in an optically accessible, fired HSDI engine at idle. The velocity field was also calculated, including the full induction stroke, using multi-dimensional fluid dynamics and combustion simulation models. A detailed comparison between the measured and calculated velocities was performed to validate the computed results and to gain a physical understanding of the flow evolution. Motored measurements were also presented, to clarify the effects of the fuel injection process and combustion on the velocity field evolution. Modification of the mean flow by fuel injection and combustion was also performed. Substantial changes in the tangential velocity near the spray path were observed during the fuel injection event and, after combustion, a radial profile of the mean tangential velocity that favours turbulence production was developed within the bowl [19]. Hountalas and Pariotis developed a simplified model for the spatial distribution of temperature in a motored direct injection diesel engine. For the fluid flow, a phenomenological model, which was coupled to a CFD method to solve the energy conservation equation and therefore the temperature field, was used. The proposed method has the advantage of simplicity and low computational time. Experiments were conducted on a direct injection diesel engine with bowl in piston at various speeds and the experimental compression curve was compared with the theoretical one. A very good agreement between the predicted and the experimental cylinder pressure was observed [20].

Dong et al. established advanced user subroutines and overall simulation strategies to model a engine in-cylinder turbulent flow, temperature, pressure, and exhaust gas recirculation (EGR) concentration fields and to simulate EGR stratification process in a typical pent-roof gasoline engine cylinder during intake and compression strokes. A series of CFD studies—combustion dome, ports, valve and valve mask configurations, EGR tolerances, swirl ratio, boundary and initial conditions, etc., had been conducted to relate the "corndog" flow structure and physical parameters—centrifugal force, heat transfer, swirl ratio, tumble flow, turbulent and burnt gas residual. Another method to stratify EGR in cylinder, named "conjugate vortex EGR stratification" system, was discovered by the CFD analysis. A clear stratification interface between the two streams of flow was found and located in the cylinder center [21]. Auriemma et al. illustrated an experimental and numerical investigation of the flow generated by an intake port model for a heavy-duty direct injection (HDDI) diesel engine. Tests were carried out on a steady-state airflow test rig to evaluate the global fluid-dynamic efficiency of the intake system, made by a swirled and a directed port, in terms of mass flow rate, flow coefficients and swirl number. In addition, the LDA technique was applied to obtain the local distribution of the air velocity within a test cylinder. The computation was carried out by the fluid-dynamic code STAR-CD that solves the ensemble averaged conservation equations for mass, momentum and energy in steady-state conditions with the turbulence model. The grid, reproducing the geometry of the intake port and the real fluid system, was made using CAD data and the boundary conditions were the same as the experimental ones. Results of the computed mass flow rate, flow coefficients, and velocity profiles was compared to the experimental ones [22].

The means of generating the initial values and boundary data was presented and the effect of different methods was discussed by Antila et al. This study deals with three different compression ignition engines with cylinder diameters of 111, 200 and 460 mm. The initial cylinder charge had been carefully analyzed through gas exchange pressure recordings and corresponding one-dimensional simulation. The swirl generated by intake ports in a high-speed engine was simulated and measured. The combustion simulation using a whole cylinder model was compared with a sector model simulation result. The difference between a sector mesh simulation and a whole cylinder model simulation was found to be considerable in case of a high-speed diesel engine. In case of a medium-speed engine the gains of using the whole cylinder model were more questionable. The simulation of fuel injection rate and injection velocity at the nozzle exit was presented in this study, as well. The validation of this simulation was made through injection pressure and injector needle lift measurements. The fuel injection boundary condition, the predicted injection velocity, was found to have a noteworthy effect on the simulated heat release [23]. Time-resolved digital particle image velocimetry (TRDPIV) data was presented for the in-cylinder flow field of a motored four-stroke, multi-valve direct injection spark ignition (DISI) optical internal combustion (IC) engine by Jarvis et al. The authors highlighted the application of a new TRDPIV system to provide both spatial and temporal in-cylinder flow field development over multiple engine cycles for improved understanding of cyclic variation in this paper [24].

Lee and Zhao modeled the blow-by phenomenon in a Small-Bore High-Speed Direct-Injection Optically Accessible Diesel Engine. They observed that the difference in peak pressure was up to 25% near TDC. To account for the pressure differences, a 0-D crevice flow model with a dynamic ring pack model was incorporated into the KIVA code to determine the amount of blow-by. The crevice flow model takes into consideration the flow through the circumferential gap, ring gap, and the ring side clearance. Chemical reactions of combustion were calculated by using the shell ignition model and KIVA's combustion subroutine. Pressure traces for each case showed dramatic improvement with

the use of the crevice flow model and reported values around 4% of blow-by has seen. One of the main causes for such high blow-by values found was attributed to the lack of an oil film layer on the surface of the cylinder wall [25].

3. Turbulence modeling

Most of computational works on diesel engines employ two-equation turbulence models, predominantly the standard k-ε model for modeling turbulence phenomena. A brief review has been made about these models.

A flow model was presented by Borgnakke et al., which predicts the swirl and turbulent velocities in an open chamber, cup-in-piston IC engine. The swirl model was based on an integral formulation of the angular momentum equation solved with as assumed tangential velocity profile form, V\gu(r). This enables the swirl model to predict a non-solid body rotation, which is a function of the inlet flow, wall shear and squish motion during the engine cycle [26].

A computer simulation for the performance of a four-stroke spark-ignition engine was used to assess the effects of in-cylinder flow processes on engine efficiency. The engine simulation model is a thermodynamic model coupled to submodels for the various physical processes of in-cylinder swirl, squish and turbulent velocities, heat transfer and flame propagation. The swirl and turbulence models, account for the effects of changes in geometry of the intake system and the chamber design on in-cylinder flow processes. The combustion model was an entrainment burn-up model applicable to the mixing controlled region of turbulent flame propagation. The flame was assumed to propagate spherically from one or two spark plug locations. A heat transfer model that was dependent upon the turbulence level was used to compute the heat loss from the unburned and burned gases. These submodels are calibrated from experimental data by Davis and Borgnakke [27]. Musculus and Rutland developed a turbulent combustion model based on the coherent flame model and applied it to diesel engines. The combustion was modeled in three distinct but overlapping phases: low temperature ignition kinetics using the shell ignition model, high temperature premixed burn using a single step Arrhenius equation and the flamelet based diffusion burn. The characteristics of the modeling approach were addressed by examining the spatial resolution of the model, results inside the engine cylinder. The location and magnitude of model heat releases, flame areas, and spray distribution were examined [28].

McLandress et al. simulated a heavy-duty caterpillar 3406 diesel engine using KIVA-3 and compared two turbulence models (k-ε and RNG k-ε). The results for the two models did not differ significantly. When the RNG k-ε model was used, the vortex in the upper far corner away from the inlet valve had a smaller radius than when the standard k-ε model was used. The large-scale fluid motion, which developed during the intake stroke, was the most significant component of mixing during the entire process. Vortex tracking elucidated the survival of several coherent large-scale structures, which caused enhanced mixing during the latter half of the compression stroke. These structure also lead to a very disordered and inhomogeneous in-cylinder flow field just prior to the time fuel would normally be injected into the engine. Two mixing parameters were very helpful in characterizing the small scale motion of turbulence. Turbulent kinematic viscosity was significant because it was the primary indicator of spatial locations which are favourable for the survival of large-scale coherent structure. The production of turbulent kinetic energy was also helpful by capturing the locations where turbulence was being created such that small-scale turbulence could be enhanced [29]. Jakirlic et al. investigated the potential of the single-point closures for predicting the flow and turbulence in piston-cylinder configurations. The study of Reynolds Averaged Navier Stokes modeling is simplified in simplified internal combustion engine configurations [30]. Bianchi et al. proposed improvements in the RNG k-ε model to consider the effect of non-equilibrium turbulence, while modeling spray combustion in diesel engines. The simulations showed that the two-equation linear eddy viscosity models failed in predicting the influence of turbulence on the fuel conservation rate. Analysis of the variation of the molecular viscosity and turbulence eddy turnover time to mean flow time scale ratio indicated that engine flows during compression, spray injection and combustion undergo non-equilibrium conditions. Three approaches had been applied in order to account for non-equilibrium turbulence. The first approach assumes that the turbulent time scale τ under non-equilibrium conditions is proportional to the turbulent Reynolds number, to account for both energy dissipation from larger to smaller scales and energy dissipation into heat at small scales. The second approach assumes that the turbulent time scale under non-equilibrium conditions is in addition proportional to the ratio between mass averaged turbulent time scale evaluated under non-equilibrium and mass averaged eddy turnover time evaluated under equilibrium. The third approach assumes that the turbulent time scale under non-equilibrium conditions is proportional to the square root of the turbulent Reynolds number, to project that there occurs a delay in response of the turbulent cascade to the changes in the mean flow. The first and second approaches improved combustion predictions over a wide range of operating conditions whereas the third was not able to show much improvement [31]. A modified version of the laminar and turbulent characteristic time combustion model and the Hiroyasu–Magnussen soot model was implemented in the flow solver Star-CD by Kaario et al. Combustion simulations of three direct injection diesel engines, utilizing the standard k-\ge turbulence model and a modified version of the RNG k-\ge turbulence model, was performed and evaluated with respect to combustion performance and emissions. Adjustments of the turbulent characteristic combustion time coefficient, which were necessary to match the experimental cylinder peak pressures of the different engines, had been justified in terms of non-equilibrium turbulence considerations. The results confirm the existence of a correlation between the integral length scale and the turbulent time scale. This correlation can be used to predict the combustion time scale in different engines. It was found that, although the standard k-\ge turbulence model produced adequate results for the larger engines, the RNG turbulence model gave better agreement with the experimental data for all engines [32].

Bianchi et al. focused on the influence of the eddy viscosity turbulence models (EVM) in CFD three-dimensional simulations of steady turbulent engine intake flows in order to assess their reliability in predicting the discharge coefficient. The results also compared with experimental measurements at the steady flow bench. High Reynolds linear and non-linear and RNG k-\ge models had been used for simulation, revealing the strong influence of both the constitutive relation and the \ge- equation formulation on the obtained results, while limits in the applicability of more sophisticated near-wall approaches are briefly discussed in the paper [33]. Miles et al. performed combined experimental and computational approach was taken to identify the source of this turbulence. The results indicated that the dominant source of the increased turbulence was associated with the formation of an unstable distribution of mean angular momentum, characterized by a negative radial gradient [34]. Bianchi and Fontanesi demonstrate that RANS simulation can provide reliable design-oriented results by using low-Reynolds cubic k-\ge turbulence

models. Different engine intake port assemblies and pressure drops had been simulated by using the CFD STAR-CD code. Computations was performed by comparing two turbulence models: low-Reynolds cubic k-\ge and high-Reynolds cubic k-\ge. The analysis leads to remarkable assessments in the definition of a correct and reliable methodology for the evaluation of engine port breathing capabilities. Comparison between numerical results and experiments showed that the low-Reynolds cubic k-\ge model was unavoidable to correctly capture the influence of port feature variations on engine permeability [35].

The one-equation subgrid scale model for the large Eddy simulation (LES) turbulence model had been compared to the popular k-\ge RNG turbulence model in very different-sized direct injection diesel engines by Kaario et al. [36]. Murad et al. compared the suitability of various turbulence models in simulating the flow. Commercial softwares (FLUENT and SWIFT) were used to compare the performance of various turbulence models. In FLUENT, a simplified vehicle model with slanted A-pillar geometry was generated using GAMBIT and in SWIFT, the simplified vehicle model was generated using Fame Hybrid. CFD simulations were carried out using FLUENT under steady-state conditions using various turbulence models (k-, k-Realize, k-RNG, k- and Spalart Allamaras). In SWIFT, k-, A-RSM and HTM2 turbulence models were used for the steady-state simulations. Investigations were carried out at velocities of 60, 100 and 140 km/h and at 0° yaw angle. Results of Cp values were compared against available experimental data. Qualitative flow visualization analyses based on the CFD results were also carried out to better understand the airflow behavior behind the A-pillar region. Results obtained using CFD model provided reasonable agreement against available experimental data [37]. Bai studied the turbulence field in an optical diesel engine operated under motored conditions using both LES and particle image velocimetry (PIV). The study was performed in a laboratory optical diesel engine based on a recent production engine from VOLVO Car. PIV was used to study the flow field in the cylinder, particularly inside the piston bowl that is also optical accessible. LES was used to investigate in detail the structure of the turbulence, the vortex cores, and the temperature field in the entire engine, all within a single engine cycle. The LES results are compared with the PIV measurements in a $40\,mm \times 28\,mm$ domain ranging from the nozzle tip to the cylinder wall. In the intake phase the large-scale swirling and tumbling flow streams are shown to be responsible for the generation of large-scale vortex pipes, which break down to small-scale, turbulent eddies. In the later phase of compression, turbulence was mainly produced in the engine bowl. Turbulence and the large-scale coherent vortex shedding due to the Kelvin–Helmholtz instability are responsible for the enhanced heat transfer between the bulk flow and the walls. A temperature in homogeneity of about 50–60 K can be generated in the cylinder [38].

PIV and paddle wheel measurements have been conducted in a static heavy-duty diesel engine rig to characterize the flow features with different valve lifts and pressure differences by Kaario et al. These measurements were compared with CFD predictions of the same engine. The simulations were done with the standard k-\ge turbulence model and with the RNG turbulence model using the Star-CD flow solver [39]. Lendormy et al. studied initial conditions for turbulence prior to combustion with the help of a four-valve, large bore diesel engine CFD model. A one-dimensional model was first used to provide boundary conditions as well as the initial flow conditions at the beginning of the simulation. Steady-state and transient boundary conditions were studied. In addition, the standard k-\ge and RNG/k-\ge turbulence models were compared. From the averaged values of turbulence kinetic energy and its dissipation rate over the cylinder volume, a re-tuned correlation for defining the initial turbulent conditions at bottom dead center (BDC) prior to the compression stroke was proposed [40].

4. Spray modeling

The simulation of evaporating droplet dynamics in diesel engine simulation are based mostly on models proposed for single droplet evaporation phenomena. A brief review of spray model has been made.

Haselman and Westbrook developed a theoretical model for calculations of the evolution of fuel injected into internal combustion engine chambers. Fuel injection in the form of a gaseous jet and in the form of a liquid droplet spray is considered. The model used the basic equations of conservation of mass, momentum, and energy in both the gaseous and liquid phases. Applications in two-dimensional symmetry of the gaseous jet form of the numerical model are described in stagnant and swirling airflows. The liquid droplet spray model, including coupling between the droplets and the gas phase medium, are described. Applications of the two-phase model are described for the case of axially symmetric injection. Finally, the liquid spray model and the gas jet model are applied to the same conditions, leading to a general assessment of the ranges of validity of the gas jet model [41].

An analytical spray model was described which can be used to calculate the penetration and trajectory of a spray in an engine combustion chamber with air swirl by Sinnamon et al. The model consists of integral continuity and momentum equations written for a steady-state gas jet. The model contains adjustable entrainment and drag parameters evaluated from experimental data. A special single-cylinder, see-through engine and a schlieren optical system were used to study transient liquid fuel sprays under varied conditions. These experimental observations were used to determine appropriate values for the adjustable parameters in the spray model. Comparisons between model calculations and the experimentally observed sprays were presented for a wide range of conditions [42]. Liu et al. modeled the effects of drop drag and breakup on fuel sprays. Spray atomization was modeled using a wave breakup model that was based on results from jet stability analysis, and also using KIVA' s Taylor analogy breakup (TAB) model. The wave model was found to give good results for both drop trajectories and breakup drop sizes. The best results were obtained with a smaller value of the breakup time model constant than previously used in spray computations. Drop trajectory and size measurement, together with high magnification photographs, indicated that drop distortion should be accounted for in sprays. Accordingly, a modified drop drag model was proposed in which the drop drag coefficient changed dynamically with the flow conditions during the drop lifetime. In the model, the value of the drag coefficient varied between the limits of a rigid sphere (no distortion) and a disk (maximum distortion). The drop distortion was computed using the Taylor analogy between a drop and a spring–mass system, and the breakup process was described using the wave model. The TAB breakup model was also found to give good predictions of drop trajectories, but the model under predicted measured the breakup drop size considerably. The TAB results were thus relatively insensitive to the drop drag model, since small drops have low inertia, and they quickly accelerated up to the gas velocity. The wave breakup and dynamic drag models were also applied to diesel sprays. The results confirmed previous studies that show that spray-tip penetration is relatively insensitive to drop breakup and drag models. However, the distribution of drop sizes within the sprays was found to be influenced by the model details. This was due to the fact that drop drag changes the drop-gas relative

velocity, and this changes the spray drop size, since the drop breakup and coalescene processes depend on the velocity. However, these changes occurred in such a way that the net effect on the spray penetration is small [43].

Senda et al. proposed new submodels for the dispersion process of the spray impinging on a flat wall to reproduce the spray/wall interaction process inside the diesel combustion chamber. These models treated the non-vaporizing spray impinging on the wall at room temperature and the vaporizing spray, which impinges on the wall at a high-temperature above the spray saturation temperature. These models were incorporated into the KIVA code to calculate the dispersion process [44]. Bai and Gosman developed and assessed a spray impingement model for diesel spray simulation. Validation of model had been performed against several sources of experimental data. The conditions investigated involved ambient and elevated pressures and temperatures, and normal and oblique impingement cases, as well as an engine. In all the cases, the predictions compared reasonably well with the experiments, and in some cases very good agreement was achieved. They concluded that the model presented was useful in predicting spray impingement events [45]. Zhang et al. introduced a very simple method to measure the liquid phase of spray in an optically accessible direct injection diesel engine. By using this method, the effects such as injection pressure, nozzle specification, intake air boost and temperature on liquid phase penetration before ignition was investigated [46]. Pelloni and Bianchi proposed a hybrid model for simulation of high pressure driven diesel fuel spray. The extensive comparison between numerical results and experimental has shown the reasonably good accuracy of the hybrid model in predicting the breakup process of high density, high pressure driven sprays. Poor prediction of spray penetration had been detected by using a grid resolution of practical internal combustion engine computations. An anisotropic model of the droplet turbulent dispersion in conjunction with the k-ε model had been presented. Significant changes in the spray structure and droplet size were detected by comparing the isotropic and the anisotropic approaches, especially near the nozzle exit [47].

Bensler et al. performed simulations modeling the spray injection of a 5-hole diesel injector in a pressure chamber. A graphical methodology was utilized to match the spray resulting from the widely used discrete droplet spray model to pressure chamber spray images. Using engine measurements to initialize the combustion chamber conditions, the compression stroke, the spray injection and the combustion simulation was performed. The novel RTZF two-zone flamelet combustion model was used for the combustion simulation and was tested for partial load operating conditions [48].

Montajir et al. investigated the fuel spray development under high injection pressure and high gas charging pressure photographically in a small direct injection diesel engine with a common rail injection system. Improvement in the spray distribution inside the cavity was attempted by changing the lip shape, side wall shape, and injection angle [49]. Bae et al. investigated the spray characteristics from diesel injectors with various nozzle geometries by microscopic and macroscopic visualization. Spray angle measured from the macroscopic images were hardly affected by the injection pressure at the atmospheric conditions. The spray angle measured from the microscopic images near the nozzle was also independent of injection pressure [50].

Moriyoshi et al. experimentally evaluated spray model for a swirl-type injector. Volume of fluid model (VOF) was used to simulate the two-phase flow inside the injector and also the liquid film formation process outside the nozzle, while discrete droplet model was used to simulate a free fuel spray in a constant volume chamber using initial conditions deduced by empirical equations or calculated results of VOF model [51]. Chryssakis and Assanis developed a comprehensive model for sprays emerging from high-pressure swirl injectors in DISI engines accounting for both primary and secondary atomization. The model considers the transient behavior of the pre-spray and the steady-state behavior of the main spray. The pre-spray modeling was based on an empirical solid cone approach with varying cone angle. The main spray modeling was based on the liquid instability sheet atomization (LISA) approach, which are extended here to include the effects of swirl [52]. An effort had been made to understand the physical phenomenon behind combustion in direct injection diesel engine and to evaluate its performance through a mathematical model by Udayakumar and Anand. The model developed consists of two phases—the spray model and the combustion model. In the spray model a 2D axis-symmetric grid was considered. The spray dispersion, penetration, and the cells that are ready for ignition at the end of ignition delay are found from the spray model. A numerical technique is used to solve the conservation equations of mass, momentum and energy in the combustion model to evaluate the properties at each crank angle. The model was also used to determine mass fractions burnt, pressure, temperature, heat release at each crank angle and performance parameters. To validate the simulation results, test had been conducted on a single-cylinder, four-stroke Kirloskar direct injection diesel engine and a reasonable agreement was found between the computed and the experimental results [53]. Strauss and Zeng found that different engines could exhibit significantly different behaviors with similar fuel sprays. This difference is attributed to the difference in scavenging flow patterns and its effect on the momentum balance between the fuel spray and the airflow. Two different sprays were tested in the fixture and in a variety of engines [54].

Lucchini et al. developed a comprehensive model for sprays emerging from high-pressure swirl injectors for gasoline direct injection (GDI) engine. The primary and secondary atomization mechanism as well as the evaporation process both in standard and superheated conditions are taken into account. The spray modeling after the injection was based on the LISA approach, modified to correctly predict the liquid sheet thickness at the breakup length. The effect of different values of the superheat degree on evaporation and impact on the spray distribution and fuel-air mixing was also analyzed [55]. Yi and DeMinco developed an engine simulation tool in a commercial CFD code to study the spray and mixing process that can be used to access the performance of a GDI engine [56]. An experimental and numerical study is conducted on the spray and mixture properties of a hole-type injector for direct injection gasoline engines. The laser absorption scattering (LAS) technique was adopted to simultaneously measure the spatial concentration distributions and the mass of the liquid and vapor phases in the fuel spray injected into a high-pressure and high-temperature constant volume vessel. The experimental results are compared to the numerical calculation results using three-dimensional CFD and the multi-objective optimization. In the numerical simulation, the design variable of the spray model was optimized by choosing spray-tip penetration, and mass of liquid and vapor phases as objective functions by Sato et al. [57]. An experimental study of real multi-hole diesel nozzles with different geometry in terms of conicity factor was performed under current direct injection diesel engines operating conditions by Payri et al. A complete characterization of the internal nozzle flow and the sprays injected under non-evaporative conditions had been performed [58]. Xue et al. developed a methodology that uses a locally refined mesh in the spray region. This methodology used an h-refinement adaptive method. In addition, they demonstrated

the efficiency and accuracy of the adaptive mesh refinement scheme [59]. Abani et al. proposed an improved spray model for reducing numerical parameter dependencies in diesel engine CFD simulations. The improved spray model was implemented in the engine simulation code KIVA-3V and was tested under various conditions, including constant volume chambers and various engine geometries with vaporizing and combusting sprays with detailed chemistry [60]. Micklow and Abdel-Salam presented a computationally efficient numerical model that accurately predicts complex spray distribution and spray penetration for a direct injection compression ignition engine configuration. A modified version of a multi-dimensional computer code KIVA-3V is used for the computations, with improved submodels for varying mean droplet diameter, varying injection velocity and drop distortion and drag [61]. Kim et al. investigated the air/fuel mixture formation and combustion characteristics in a spray-guided GDI engine using a commercial code, STAR-CD. The spray injection has modeled arranging multiple points using random function along the ring-shaped nozzle exit. The present approach reasonably predicted the spray behavior, the mixture distribution near the spark plug, and flame propagation inside the combustion chamber [62].

5. Conclusions

Improved computational mesh generation techniques and efficiency of solver have widely influenced the application of CFD methods to reciprocating engine models, which require movable domain boundaries and compressible/expandable meshes. Multi-dimensional models are able to predict in-cylinder mean gas velocities with high accuracy, but RMS components are predicted with lesser accuracy. The in-cylinder flow measurements are performed with non-intrusive methods such as LDA, PIV and optical methods, thus increasing the reliability of experimental data obtained inside the engine cylinder. Most computational works on diesel engines employ two-equation turbulence models, predominantly the standard k-ε model for modeling turbulence phenomena. The simulation of evaporating droplet dynamics in diesel engine simulations is based mostly on models proposed for single droplet evaporation phenomena. Ignition models used are based on the shell ignition model. A large number of combustion models are constituted for diesel combustion simulation, but do not find further validation. The laminar and turbulent characteristics time scale model finds higher usage when compared to the other models for simulating diesel combustion.

Most combustion simulations are concentrated in the mid and high load range. However, given the importance of higher emission weighting factors at idling conditions in various duty cycle, and until recent, the availability of considerable amount of in-cylinder experimental data obtained under firing conditions at low loads, work on engine simulations are not available at low speeds.

References

[1] Margery, R, Nino, E, Vafidis, C. The effect of inlet duct length on the in-cylinder air motion in a motored diesel engine. SAE Paper No. 900057.
[2] Zhang L, Minami T, Takatsuki T, Yokota K. An analysis of the combustion of a DI diesel engine by photograph processing. SAE Paper No. 930594.
[3] Chen A, Dent JC. An investigation of steady flow through a curved inlet port. SAE Paper No. 940522.
[4] Kong S-C, Ricart LM, Reitz RD. In-cylinder diesel flame imaging compared with numerical computations. SAE Paper No. 950455.
[5] Zhang L, Ueda T, Takatsuki T, Yokota K. A study of the effects of chamber geometries on flame behaviour in a direct injection diesel engine. SAE Paper No. 952515.
[6] Stephenson PW, Rutland CJ. Modeling the effects of valve lift profile on intake flow and emissions behavior in a direct injection diesel engine. SAE Paper No. 952430.
[7] Stepheson PW, Claybaker PJ, Rutland CJ. Modeling the effects of intake generated turbulence and resolved flow structures on combustion in direct injection diesel engine. SAE Paper No. 960634.
[8] Catania AE, Spessa E. Speed dependence of turbulent properties in a high squish automotive engine combustion system. SAE Paper No. 960268.
[9] Leylek J, Taylor W, Tran LT, Ronald DS. Advanced computational methods for predicting flow losses in intake regions of diesel engines. SAE Paper No. 970639.
[10] Bo T, Clerides D, Gosman AD, Theodossopoulos P. Prediction of flow and spray processes in an automobile direct injection diesel engine. SAE Paper No. 970882.
[11] Chen A, Veshagh A, Wallace S. Intake flow predictions of a transparent direct injection diesel engine. SAE Paper No. 981020.
[12] Auriemma M, Corcione FE, Macchioni R, Valentino G. Interpretation of air motion in reentrant bowl in-piston engine by estimating Reynolds stresses. SAE Paper No. 980482.
[13] Beard P, Mokaddem K, Baritaud T. Measurement and modeling of the flow-field in a direct injection diesel engine: effects of piston bowl shape and engine speed. SAE Paper No. 982587.
[14] Brunt MFJ, Kieron CP. Calculation of heat release in direct injection diesel engines. SAE Paper No. 1999-01-0187.
[15] Okazaki T, Miyazaki H, Sugimoto M, Yamada S, Aketa M. CFD approach for optimum design of direct injection combustion system in small versatile diesel engine. SAE Paper No. 1999-01-3261.
[16] Bianchi GM, Richards K, Reitz RD. Effects of initial conditions in multidimensional combustion simulations of HSDI diesel engines. SAE Paper No. 1999-01-1180.
[17] Lin L. Effects of combustion chamber geometry on in-cylinder air motion and performance in DI diesel engine. SAE Paper No. 2000-01-0510.
[18] Reitz RDR, Miles PCM, Sick V, Nagel Z, Richards K, Megerle M. The evolution of flow structures and turbulence in a fired HSDI diesel engine. SAE Paper No. 2001-01-3501.
[19] Miles PCM, Sick V, Megerle M, Richards K, Nagel Z, Reitz RD. The evolution of flow structures and turbulence in a fired Hsdi diesel engine. SAE Paper No. 2001-01-3501.
[20] Hountalas DT, Pariotis EG. A simplified model for the spatial distribution of temperature in a motored Di diesel engine. SAE Paper No. 2001-01-1235.
[21] Dong M, Chen GT, Xu M, Daniels CF. A preliminary Cfd investigation of in-cylinder stratified Egr for spark ignition engines. SAE Paper No. 2002-01-1734.
[22] Auriemma M, Caputo G, Corcione FE, Valentino G, Riganti G. Fluid-dynamic analysis of the intake system for a Hddi diesel engine by Star-Cd code and Lda technique. SAE Paper No. 2003-01-0002.
[23] Antila E, Larmi M, Saarinen A, Tiainen JT, Kaario OT, Laaksonen M. Cylinder charge, initial flow field and fuel injection boundary condition in the multi-dimensional modelling of combustion in compression ignition engines. SAE Paper No. 2004-01-2963.
[24] Jarvis S, Justham T, Clarke A, Garner CP, Hargrave GK, Richardson D. Motored SI IC engine in-cylinder flow field measurement using time resolved digital PIV for characterisation of cyclic variation. SAE Paper No. 2006-01-1044.
[25] Lee C-FF, Zhao J. Modeling of blowby in a small-bore high-speed direct-injection optically accessible diesel engine. SAE Paper No. 2006-01-0649.
[26] Borgnakke C, Davis GC, Tabaczynski RJ. Predictions of in-cylinder swirl velocity and turbulence intensity for an open chamber cup in piston engine. SAE Paper No. 810224.
[27] Davis GC, Borgnakke C. The effect of in-cylinder flow processes (swirl, squish and turbulence intensity) on engine efficiency—model predictions. SAE Paper No. 820045.
[28] Musculus MP, Rutland CJ. An application of the coherent flamelet model to diesel engine combustion. SAE Paper No. 950281.
[29] McLandress A, Emerson R, McDowell P, Rutland C. Intake and in-cylinder flow modeling characterization of mixing and comparisons with flow bench results. SAE Paper No. 960635.
[30] Jakirlic S, Hadzic I, Pascal H, Hanjalic K, Tropea C. Computational study of joint effects of shear, compression and swirl on flow and turbulence in a valve less piston–cylinder assembly. SAE Paper No. 2001-01-1236.
[31] Bianchi GM, Pelloni P, Zhu G, Reitz RD. On non-equilibrium turbulence corrections in multidimensional HSDI diesel engine computations. SAE Paper No. 2001-01-0997.
[32] Kaario O, Tanner F, Larmi M. Relating integral length scale to turbulent time scale and comparing K–Ge and Rng K–Ge turbulence models in diesel combustion simulation. SAE Paper No. 2002-01-1117.
[33] Bianchi GM, Fontanesi S, Cantore G. Turbulence modelling in Cfd simulation of ice intake flows: the discharge coefficient prediction. SAE Paper No. 2002-01-1118.
[34] Miles P, Megerle M, Hammer JS, Nagel Z, Reitz RD, Sick V. Late-cycle turbulence generation in swirl-supported, direct-injection diesel engines. SAE Paper No. 2002-01-0891.
[35] Bianchi GM, Fontanesi S. On the applications of low-Reynolds cubic K–Ge turbulence models in 3d simulations of ice intake flows. SAE Paper No. 2003-01-0003.
[36] Kaario O, Pokela H, Kjäldman L, Tiainen J, Larmi M. Les and Rng turbulence modelling in Di diesel engines. SAE Paper No. 2003-01-1069.
[37] Murad NM, Naser J, Alam F, Watkins S. Simulation of vehicle A-pillar aerodynamics using various turbulence models. SAE Paper No. 2004-01-0231.

[38] Bai X-S. Numerical and experimental investigation of turbulent flows in a diesel engine. SAE Paper No. 2006-01-3436.
[39] Kaario O, Lendormy E, Sarjovaara T, Larmi M, Rantanen P. In-cylinder flow field of a diesel engine. SAE Paper No. 2007-01-4046.
[40] Lendormy E, Kaario O, Larmi M. Three-dimensional CFD modeling of the initial turbulence prior to the compression stroke in a large bore diesel engine. SAE Paper No. 2008-01-0977.
[41] Haselman LC, Westbrook CK. A theoretical model for two-phase fuel injection in stratified charge engines. SAE Paper No. 780318.
[42] Sinnamon JF, Lancaster DR, Steiner JC. An experimental and analytical study of engine fuel spray trajectories. SAE Paper No. 800135.
[43] Liu AB, Mather D, Reitz RD. Modeling the effects of drop drag and breakup on fuel sprays. SAE Paper 930072.
[44] Senda J, Kobayashi M, Iwashita S, Fujimoto H. Modeling of diesel spray impingement on a flat wall. SAE Paper No. 941894.
[45] Bai C, Gosman AD. Development of methodology for spray impingement simulation. SAE Paper No. 950283.
[46] Zhang L, Yokota K, Ueda T, Ishii Y, Itou T, Tsurusima T, et al. Measurement of liquid phase penetration of evaporating spray in a direct injection diesel engine. SAE Paper No. 971645.
[47] Pelloni P, Bianchi GM. Modeling the diesel fuel spray breakup by using a hybrid model. SAE Paper No. 1999-01-0226.
[48] Bensler HP, Buehren F, Vervisch L. 3-D Cfd analysis of the combustion process in a Di diesel engine using a flamelet model. SAE Paper No. 2000-01-0662.
[49] Montajir RMd, Tsunemoto H, Ishitani H, Minami T. Effects of reverse squish on fuel spray behaviour in a small DI diesel engine under high pressure injection and high charging conditions. SAE Paper No. 2000-01-2786.
[50] Bae C, Yu J, Kang J, Kong J, Lee KO. Effect of nozzle geometry on the common rail diesel spray. SAE Paper No. 2002-01-1625.
[51] Moriyoshi Y, Hu X, Takagi M. Experimentally evaluated spray model for a swirl-type injector. SAE Paper No. 2002-01-2696.
[52] Chryssakis C, Assanis DN. Fuel spray simulation of high-pressure swirl injector for Disi engines and comparison with laser diagnostic measurements. SAE Paper No. 2003-01-0007.
[53] Udayakumar R, Anand MS. Multizone combustion model for a four-stroke direct injection diesel engine. SAE Paper No. 2004-01-0921.
[54] Strauss S, Zeng Y. The effect of fuel spray momentum on performance and emissions of direct-injected, two-stroke engines. SAE Paper No. 2004-32-0013.
[55] Lucchini T, D'Errico G, Nordin N. CFD modelling of gasoline sprays. SAE Paper No. 2005-24-086.
[56] Yi Y, DeMinco CM. Numerical investigation of mixture preparation in a GDI engine. SAE Paper No. 2006-01-3375.
[57] Sato K, Nishida K, Tadokoro T, Yamakawa M, Yokohata H. Spray and mixture properties of hole-type injector for DI gasoline engine—comparison of experiment and CFD simulation. SAE Paper No. 2007-01-1850.
[58] Payri F, Payri R, Salvador FJ, Gimeno J. Influence of nozzle geometry on spray characteristics in non-evaporative and evaporative conditions. SAE Paper No. 2007-24-0023.
[59] Xue Q, Kong S-C, Torres DJ, Xu Z, Yi J. DISI spray modeling using local mesh refinement. SAE Paper No. 2008-01-0967.
[60] Abani N, Kokjohn S, Park SW, Bergin M, Munnannur A, Ning W, et al. An improved spray model for reducing numerical parameter dependencies in diesel engine CFD simulations. SAE Paper No. 2008-01-0970.
[61] Micklow G, Abdel-Salam TM. Fuel spray modeling for direct injection compression ignition engine configurations. SAE Paper No. 2008-01-0947.
[62] Kim S-J, Kim Y-N, Lee J-H. Analysis of the in-cylinder flow, mixture formation and combustion processes in a spray-guided GDI engine. SAE Paper No. 2008-01-0142.